REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. REPORT DATE	2. REPORT TYPE	3. DATES COVERED
	Professional Paper	

4. TITLE AND SUBTITLE

Time Accurate Computational Simulations of Ship Air Wake

5a. CONTRACT NUMBER

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER

6. AUTHOR(S)

Susan Polsky, Chris Bruner

5d. PROJECT NUMBER

5e. TASK NUMBER

5f. WORK UNIT NUMBER

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)

Naval Air Warfare Center Aircraft Division
22347 Cedar Point Road, Unit #6
Patuxent River, Maryland 20670-1161

8. PERFORMING ORGANIZATION REPORT NUMBER

9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES)

Naval Air Systems Command
47123 Buse Road Unit IPT
Patuxent River, Maryland 20670-1547

10. SPONSOR/MONITOR'S ACRONYM(S)

11. SPONSOR/MONITOR'S REPORT NUMBER(S)

12. DISTRIBUTION/AVAILABILITY STATEMENT

Approved for public release; distribution is unlimited.

13. SUPPLEMENTARY NOTES

14. ABSTRACT

Time accurate computational fluid dynamics (CFD's) calculations were performed to characterize the unsteady nature of the air wake produced by a U.S. Navy LHA class ship. A series of wind-over-deck (WOD) conditions were examined. It was found that the general character of the air wake changed dramatically as the WOD angle changed. The air wake was also found to be highly unsteady or "turbulent" but also contained some strongly periodic features. The CFD simulations were done at full-scale conditions with second order time accuracy using the COBALT CFD code. COBALT is an unstructured grid, Navier-Stokes solver.

15. SUBJECT TERMS

computational fluid dynamics (CFD's)

16. SECURITY CLASSIFICATION OF: a. REPORT	b. ABSTRACT	c. THIS PAGE	17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
					Susan Polsky / Chris Bruner
					19b. TELEPHONE NUMBER (include area code)
Unclassified	Unclassified	Unclassified	Unclassified	5	(301) 342-8575 / 8542

Abstract for

Time Accurate Computational Simulations of Ship Air Wake

Susan A. Polsky and Chris W. S. Bruner

Time accurate computational fluid dynamics (CFD) calculations were performed to characterize the unsteady nature of the air wake produced by a US Navy LHA class ship. A series of wind-over-deck (WOD) conditions were examined. It was found that the general character of the air wake changed dramatically as the WOD angle changed. The air wake was also found to be highly unsteady or "turbulent" but also contained some strongly periodic features. The CFD simulations were done at full-scale conditions with second order time accuracy using the COBALT CFD code. COBALT is an unstructured grid, Navier-Stokes solver.

Introduction

The air wake produced by the superstructure, deck, etc. of ships used for aircraft/rotory wing operations is of continuing interest to the US Navy as well as to other DoD services. Air wake can have a significant impact on aircraft operations and much time and money is spent developing allowable wind-over-deck (WOD) operating envelops for each airvehicle which must use the ship as a platform. The ability to predict the air wake over a ship can aid in the development of aircraft operating envelopes and can be used to "diagnose" problems that cause some landing spots to be problematic or unusable.

To date, wind tunnels have been used almost exclusively to predict ship air wake. The draw back of this approach is two fold. The first issue involves the size and speed of real ships in comparison with the available size and speed ranges of wind tunnels. Because the ships in questions are so large, the scale factors required to build reasonably sized wind tunnel models is generally in the $1/100^{th}$ range. Because of this, the speeds required in a wind tunnel to match full scale Reynolds numbers are unattainable. This introduces Reynolds number and scaling issues when attempting to scale back to full scale values. It can be argued that these flows are largely Reynolds number independent (this will be discussed in more detail later); however, this has not been shown conclusively, and the question of scale is never far from the forefront. The second drawback to wind tunnel testing has to do with the complexity of the flow field all around the ship. There are large flow features such as vorticies and separations that are major drivers in the flow. Although much can be learned from surface measurements, off-body flow field characterization is an important part of analyzing and understanding ship air wake. Unfortunately in wind tunnels it is often both expensive and difficult to obtain quantitative off body flow field measurements. Computational fluid dynamics (CFD) has been used in this work as an alternative approach to predicting ship air wake. CFD can be used to compute full-scale solutions and to obtain detailed off body and time accurate data.

Numerical Method

The CFD solver COBALT (ref) was used for this study. COBALT is an unstructured-grid, Navier-Stokes solver that is optimized to run in a parallel environment. The code was run in laminar mode. In other words, no traditional one- or two-equation turbulence model was applied; however, the turbulence was modeled using Monotone Integrated Large Eddy Simulation (MILES) (ref). MILES turbulence modeling is a form of Large Eddy Simulation (LES) which is inherent in time-accurate, flux-limiting schemes such as the one used in COBALT. The code was run with second order accuracy in both time and space. The time step used (generally 0.01 seconds) was chosen to resolve frequencies up to 40 Hz based on the Niquist Criteria.

Unfortunately, COBALT, which is a density-based code, does not have a preconditioner to compensate for the incompressible flow regime being examined. However, as is demonstrated in the "Validation" and "Velocity Independence" sections below, this does not seem to have adversely affected the solution.

Unstructured, hybrid grids were generated using a combination of VGRID(ref) and Blacksmith(ref). VGRID constructs grids that have purely tetrahedral cells. Blacksmith recombines the tetrahedrons in all the complete viscous layers into prisms. Thus, the hybrid grids consist of tetrahedrons in the inviscid regions and prisms in the viscous layers. Before the tetrahedral cells in the viscous layers were recombined into prisms, the overall grid sizes were in the 7 million cell range. After recombining the tetrahedrons, the overall grid sizes were in the 4 million cell range. The largest cell on the ship deck was approximately 5 ft across. The outer boundaries of the grid volume were two ship lengths in front and to either side of the ship and three ship lengths behind the ship. The solutions were started impulsively.

The solutions were run on an IBM SP P3 using 64 processors. Each solution required at lease 4000 iterations and approximately 192 hours of wall clock time.

Results

The geometry for the ship, an LHA class Navy ship, is shown in Fig. 1. A picture of the actual ship is shown in Fig 1a and the CFD model is shown in Fig. 1b. There is considerable detail modeled including the major features of the island, the lip around the deck edge, and the deck edge elevator. The main features that have been neglected are the "yellow gear" (equipment used in daily operations) and the antenna arrays on the island. The yellow gear was neglected mainly because the various pieces of equipment are relatively small (compared to the size of the ship) and because they are not in fixed positions. The antenna arrays were neglected because it was assumed that they were highly porous and would not have significant influence on the flow. The complexity of modeling the antenna geometry was also a factor in this decision. In future work, it would be desirable to add the antenna arrays to the island to quantify the effect on the flow field.

Validation

Validation of CFD simulations when a code is applied to a new class of problems is always an important step. In this case, there is no full-scale data to compare with; therefore, wind tunnel data is used. The model and the plane in which data was taken are shown in fig. 2. The test article is a simplified version of an LHA at $1/120^{th}$ scale. The data was non-dimensionalized by the tunnel speed, 150 ft/s. A seven hole probe was used to collect u,v, and w components of velocity. It should be noted that although the flow is unsteady, the seven hole probe will essentially record a steady, time-averaged value of velocity. Because the test article caused a significant amount of blockage in the tunnel, the CFD simulation modeled the tunnel walls.

A time accurate solution was obtained and the velocity values were averaged over 0.022 seconds in order to compare directly with the seven hole probe data. The 0.022 second time period consisted of 1100 time steps of $2x10^{-5}$ seconds each. Comparisons of the wind tunnel and CFD data for increasing heights off the deck are shown in figs. 3a-3c. In these figures, x runs longitudinally down the ship, y runs across the ship from starboard to port, and z runs from the deck surface up. A y-value of 0 indicates the centerline of the ship. The data compares very well, and the major flow features such as the two deck edge vorticies are clearly indicated in the CFD data. Note that the data is not symmetric about y=0. This is due to the upstream influence of the island. The unsteady nature of the u-velocity component is demonstrated in fig. 3a where, in addition to the time-averaged u-velocity, the standard deviation in time of the u-velocity is also plotted.

Full-scale Calculation

Reynolds Number Independence

The effects of Re number are currently being studied. The results will be presented in the full paper.

WOD Speed Independence

For both computational and practical reasons, it may be desirable to artificially increase the WOD speed. It is important to note that only the speed and, therefore, the Mach number are changed. The Re number remains constant. This is accomplished by changing the density and pressure.

On the computational side, it is desirable to increase the speed (and thus the Mach number) to aid in convergence or, in unsteady cases such as these, to decrease the number of iterations required to establish the flow. In addition, increasing the Mach number may reduce potential numerical errors due to the incompressible nature of the flow. On the practical side, if solutions are desired for the same wind angle but several different wind speeds, one solution could be computed and then scaled to the other desired speeds.

In order to explore this option, two solutions were computed with 15 and 30 knt wind speeds. The wind angle was held constant at 330 degrees. The solution from the 30 knt case was then scaled down to 15 knt simply be dividing the velocity components by a factor of 2. Comparisons between the two cases were made along the ship centerline, along a typical approach path and across several landing spots. For all the locations examined, the profiles compared very well. As an example, ship centerline comparisons between the 15 knt and the scaled 30 knt cases are shown in fig. 4.

It is important to note that this technique is limited in its range of application. If the speed is scaled too high, compressibility effects will corrupt the solution. Similarly, as the speed approaches zero, obviously the flow will look very different that it does at 15 kts. Therefore, a minimum of two speeds must always be computed (the maximum and minimum speeds of interest) before scaling can be performed with confidence.

LHA Simulations

The full scale LHA model is show in fig. 1b. A series of 7 WOD conditions were computed for wind angles of 0-90 and 270-360 at every 30 degrees (fig. 5), and for wind speeds of 15 and 30 knts. Only a portion of these conditions will be presented here. A total of 50-70 seconds of time was computed for each case. After examining several cases, it was determined that after 40 seconds had elapsed the flow was well established and all transients due to the impulsive had flushed out of the domain. Forty seconds was chosen based on two criteria. First, after 40 seconds, the volume of air over the ship will have flushed through 2.5 times allowing the flow to become well established. The second criterion was based on examining periodic trends in surface pressure and off-body velocity data. Once some obvious feature(s), usually some shedding event, had repeated itself several times, it was assumed that the flow was well established. The remaining 10-30 seconds of data was used for analyses.

The first case presented is for a WOD of 30kts and 0 degrees. Surface pressure contours at an instant in time are shown in fig. 6. Much can be learned about the flow based on this figure such as the existence of a large separation at the bow indicated by the distinct low pressure region (shown in blue). However, the truly complex nature of the flow field is not revealed until the off body flow is examined. Iso-surfaces of vorticity are shown in fig. 7. Again, only more clearly, the separation off the bow is shown. In addition deck edge vorticies are easily seen. What is further revealed, however, are "doughnut" shaped features between the bow-separation reattachment point and the island. In addition, the highly complex nature of the flow around and behind the island is indicated. Vorticity along the deck centerline plane in shown in Fig. 8. Here again the "doughnuts" are easily seen and the overall complexity of the flow is demonstrated. When the solution was animated, it was found that the "doughnuts" are shed periodically from the bow separation at a frequency of approximately 2-3Hz. In addition, it appears that the shedding from the bow separation excites the deck edge vorticies causing them to "bulge". The bulge in the vortex persists downstream until it is disturbed by geometric features or other flow features. The doughnuts impinge on the island and interact with the island separation. All this feeds back into the area behind the island where the flow is very complicated. In general, the flow behind the island is very unsteady and, thus far, no periodic features have been seen in this region.

The next case is again for a wind speed of 30kts; however, the wind direction is now 30 degrees. Comparing surface pressures for the 30deg case (fig. 9) and those for the 0 deg case (fig. 6) it can be seen that the major flow features change dramatically. First the separation off the bow is almost non-existent in the 30deg case. In addition, there is a significant separation off the starboard deck edge and considerably more of the island is separated (all indicated by low pressure (blue) areas on the surface). Again, iso-surfaces of vortcity in the off-body flow demonstrate the differences much more clearly (fig 10). The bow separation has been blown off the front of the ship. Since this separation was the driver to create the periodic shedding of the "doughnuts", these features do not appear in this flow. It is also shown that what appeared as a separation along the starboard deck edge is actually the deck edge vortex being blow across the deck. In addition, the port side deck edge vortex is detached from the deck edge and blown out over the ocean. The flow behind the island is more dramatically separated, and there does seem to be some periodic shedding off the island itself; however, this has not yet been confirmed by animating the solution.

Conclusions

The air wake generated by an LHA class ship is highly unsteady, but also contains features that are shed periodically. In addition, relatively stable features exists such as the deck edge vorticies and the bow separation. However, the steady features can be influenced by large periodic features and by large "random" features. All of these features and interactions may influence flight operations especially nearby and behind the island where the flow appears to be very chaotic.

CFD has become a practical tool for predicting ship air wake flowfields. The advantages of using CFD include the ability to model the full-scale aerodynamics, the level of detail obtainable for both surface data and off-body data, and the ability to collect and animate time accurate data to reveal flow features that would have otherwise been missed.

Future Work

For future work, we intend to fill in the WOD ranges to every 15 degrees, to explore the influences of a helicopter hover over a landing spot, to perform a grid sensitivity study and to include the influences of the antenna arrays.

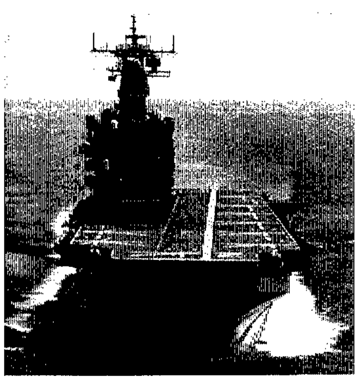

Fig 1a. LHA 2, USS Saipan

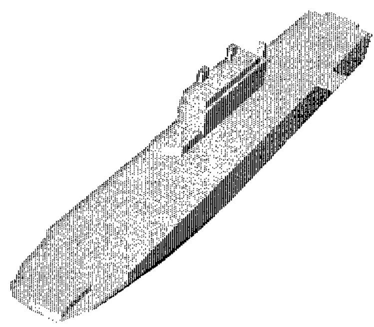

Fig 1b. CFD Model of LHA 2

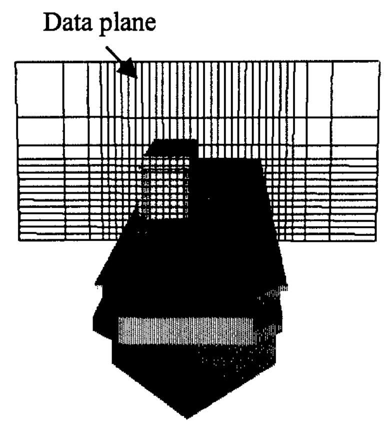

Fig 2. Windtunnel Model of LHA

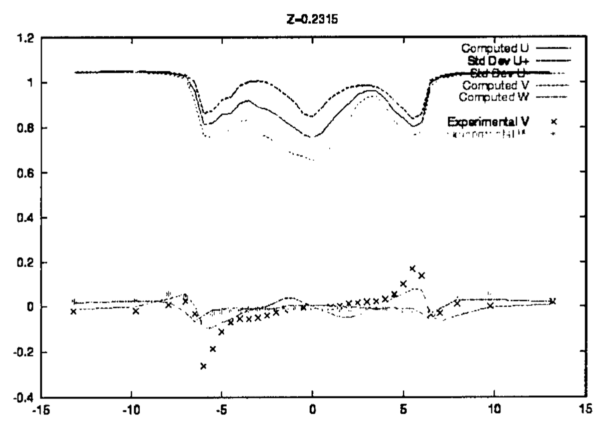

Fig 3a. Comparison of Windtunnel and CFD Data, Z=0.2315in

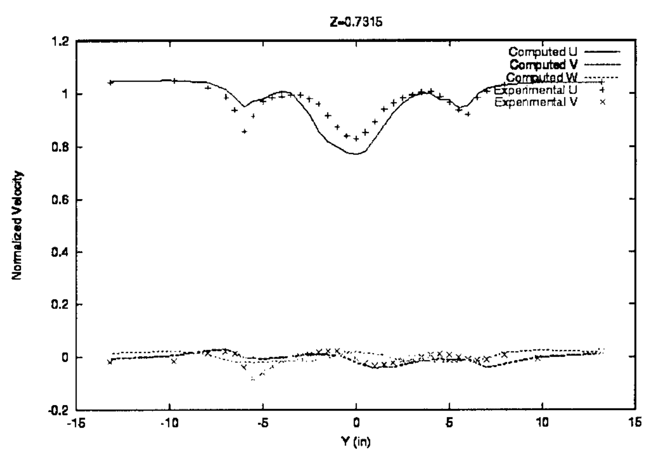

Fig 3b. Comparison of Windtunnel and CFD Data, Z=0.7315in

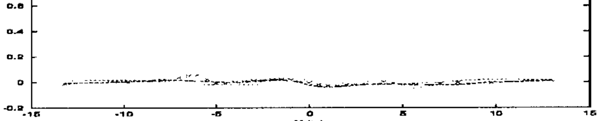

Fig 3c. Comparison of Windtunnel and CFD Data, Z=1.3in

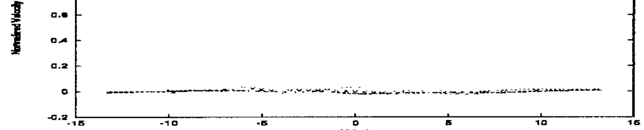

Fig 3d. Comparison of Windtunnel and CFD Data, Z=2.3in

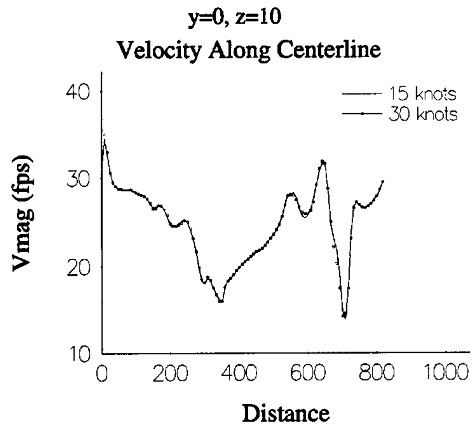

Fig 4. Comparison of 15kt and scaled 30kt velocity magnitude alon the ship centerline

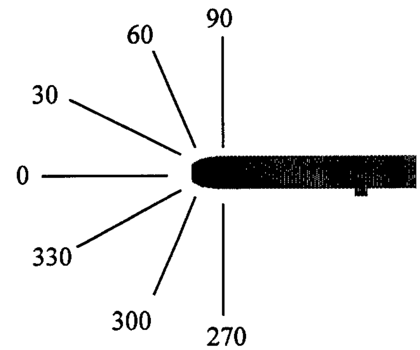

Fig 5. Wind-over-deck azimuths

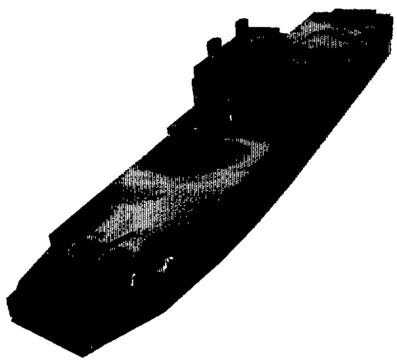

Fig 6. Surface pressure for WOD=0°,30kt

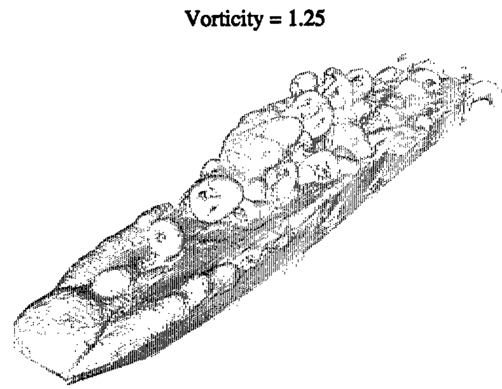

Fig 7. Vorticity iso-surfaces for WOD=0°,30kt

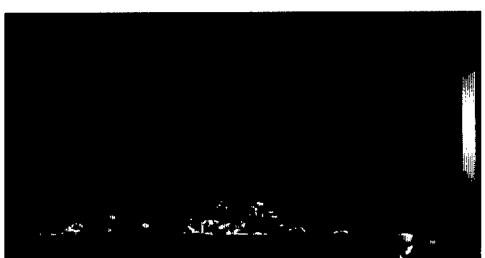

Fig 8. Vorticity magnitude on the ship centerline for WOD=0°,30kt

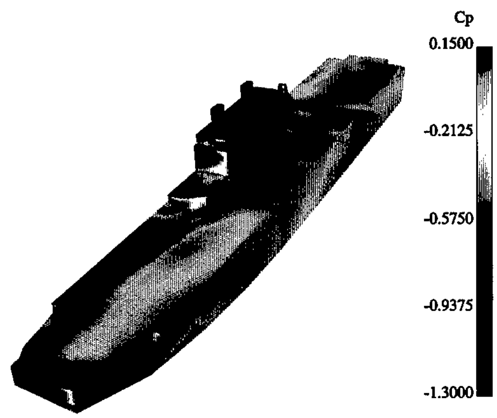

Fig 9. Surface pressure for WOD=30°,30kt

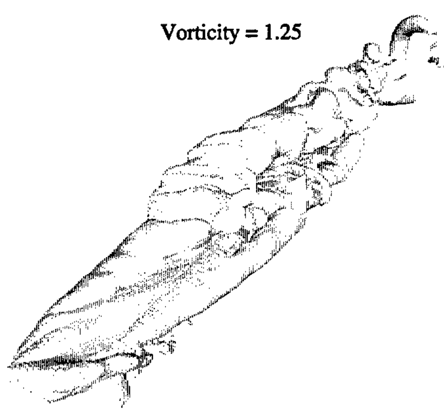

Fig 10. Vorticity iso-surfaces for WOD=30°,30kt